Structural analysis with mapping of formability analysis results for sheet metal part in automotive application

#1Rahul R. Bhore, #2Prof. Amol B. Gaikwad

1 bhorerahul@gmail.com
2 amol.gaikwad@dypic.in

1Dr. D. Y. Patil School of Engineering
Lohgaon, Pune, India.

ABSTRACT
Deep drawing is most important sheet metal forming process for majority of automobile components. In sheet metal forming, material undergoes permanent deformation, this deformation mechanism is complicated and final mechanical properties are very difficult to predict. Traditional design methods for sheet metal forming components are usually based on trial and error or empirical approach. Due to growing demand of high precision and reliability for formed metal part, advances in FEA is being widely adopted. Formability simulation concerns with investigation & understanding of deformation mechanism and used to evaluate effect of various process parameters at the product design stage. Manufacturing effects from sheet metal forming process can strongly affect structural response ranging from crashworthiness to durability. Deep drawing typically results in significant strain hardening; thinning and residual stresses in formed part, so advanced computational process are required to consider these forming-induced effects into structural analysis. To increase accuracy and reliability of structural finite simulations, it is essential to map formability analysis results to depict more realistic structural integrity.

In this paper study performed for a deep drawing component of passenger car drive shaft assembly. Altair’s HyperForm Radioss 1-Step is used for formability analysis and predicting formability and other results like %thickness, major-minor stains and effective stress. HyperWorks Result Mapper is HyperCrash based pre-processing tool is used for mapping formability analysis results to structural model. Mapping quantities include thickness, plastic strain, strain tensor and stress tensor. Radioss/ Abaqus are used for structural analysis with mapped results from formability analysis.

Keywords— Design Engineering, Deep Drawing, Finite Element Analysis, Formability Analysis, Result mapping, Structural analysis with mapped data.

I. INTRODUCTION
The process of new product development through sheet metal forming is one of the most common manufacturing processes existing in industries; however it is not easy to design the process and the tools for most of the components. Material breakages, shape defects, thinning, wrinkling, are
the main defects encountered in sheet metal forming operation. Such defects can be foreseen during product feasibility study using FEA forming simulation technique, and through optimized design changes & process parameters, their occurrence can be prevented during manufacturing.

Sheetmetal forming processes have been carried out with trial and error experimental work in the traditional methods without complete understanding of the complicated deformation mechanism and plasticity theory. The finite element method has been introduced to the analysis of the forming process and has provided useful information for optimum design of process parameters which results in decrease the amount of strain that prevents fracture by tearing. Since the deformation mechanism is very complicated and the final mechanical properties are difficult to predict, it is not easy to develop a product with desired shape and final material properties to serve the intended applications. The deformation inherently proceeds with the irregular shapes of the cross-section and conditions that the cause failure such as excessive thinning, tearing and wrinkling. Success or failure of the forming process is influenced by many parameters such as the blank holding force, drawing ratio, the shape of the die, strain-hardening coefficient, and lubrication conditions [1]. Thus, formability analysis for the investigation and understanding of deformation mechanics has become a major concern in product feasibility & development.

In large automotive enterprises, forming simulations and functional performance analyses ranging from crash to durability to NVH are generally performed by separate groups with limited data sharing and re-use. With widely varying engineering objectives, these groups adopt different software tools that use disparate mesh densities, mesh patterns and element formulations. Usually, detailed numerical simulation of a sheet-forming process requires highly refined meshes, whereas the model used for subsequent structural analysis of the same components may require somewhat coarser meshes. Consequently, to consider forming effects for subsequent structural analysis, forming results such as thickness variations, residual stresses and plastic strains must be correctly and efficiently mapped into relevant models [2].

The study of optimization of parameters in deep drawing for component is performed with the help of predictive tool Altair HyperForm Radioss. To determine the optimum values of the process parameters, it is essential to determine their influence on the deformation behaviour of the sheet metal. The most important process parameters for affecting thickness distribution are blank holding force, punch speed and friction coefficient were determined.

II. PROBLEM DEFINATION

A component taken for formability and subsequent structural analysis is Boot Can, a part of light passenger vehicle driveshaft assembly. Company supporting this work is engaged is the world’s leading provider of efficient axle & driveshaft solutions and offering innovative design solution. According to design requirements, final component should not have thinning less than 1.3mm where initial blank thickness is 2mm. So, while the part could be produced by actual try-out using the die and punch tool set, the product feasibility study is required for predicting formability & % thinning. The optimize set of process parameter need to evaluate through FEA virtual tryouts to reduce no of actual try-outs, which helps to reduce lead time for component development & cost associated with it. The functional needs of component should not be affected by the manufacturing-induced effects or residual stresses, so its structural analysis for intended purposes needs to consider forming results.

Below are the main objectives to be pursued while attempting this work:

- Fast and Accurate Feasibility study: predicting formability and other results like % thinning, major & minor effective stains and different stress results.
- The accurate blank shape prediction to minimize material scrap at the early stages of the product development process.
- Validate stamping operation in virtual try-outs with optimized process parameter.
- Results Mapping: It enables rapid initialization of structural CAE models with thinning and work hardening resulting from stamping in order to incorporate the effect of manufacturing on structural performance.
- Structural analysis with result mapping of formability analysis to improve the accuracy and reliability of structural analysis results.

III. PROCESS METHODOLOGY

A. Forming Analysis: General approach in sheet metal forming FEA involves following steps:

- Geometric definition- Preparing the model
  The sheet metal part which is to be formed is taken into HyperForm as an IGES data and model is prepared for pre-processing. It consist Geometry clean-up, Extraction of mid-surface, meshing the mid-surface.
- Model discretization/ Pre-processing:
  Using the available features in HyperForm model is prepared to run analysis. It consist Assigning the material properties and thickness to the part, selecting the blankholder/ drawbeads & stamping direction.

Fig. 1 Boot Can in driveshaft assembly & loading conditions

Fig. 2 Assigning the blankholder properties
### TABLE I
PROPERTIES FOR ASSIGNED MATERIAL

<table>
<thead>
<tr>
<th>Sr. No.</th>
<th>Property</th>
<th>Notation in HyperForm</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Young’s Modulus</td>
<td>E</td>
<td>210000</td>
</tr>
<tr>
<td>2</td>
<td>Poisson’s Ratio</td>
<td>nu</td>
<td>0.3</td>
</tr>
<tr>
<td>3</td>
<td>Tensile Strength</td>
<td>Ts</td>
<td>276 Mpa</td>
</tr>
<tr>
<td>4</td>
<td>Yield Strength</td>
<td>Ys</td>
<td>138 Mpa</td>
</tr>
<tr>
<td>5</td>
<td>Strain Hardening Exponent</td>
<td>n</td>
<td>0.26</td>
</tr>
<tr>
<td>6</td>
<td>Pre-Strain Coefficient</td>
<td>C0</td>
<td>0.00665</td>
</tr>
<tr>
<td>7</td>
<td>Strength Coefficient</td>
<td>K</td>
<td>508.081</td>
</tr>
<tr>
<td>8</td>
<td>Plastic Strain Ratio</td>
<td>r</td>
<td>1.9</td>
</tr>
</tbody>
</table>

- **Solution Generation – Solving/Processing**
  This part is done by solver automatically at backend. Several iterations were performed. The study for different coefficients friction, blank holding force and pressure level are performed with simulation solver HyperForm-One Step Radioss.

- **Analysis of Results - Post-Processing**
  Results of simulation are interpreted. Based on observation, changes are made in the process and tool-build parameters. Several iterations are carried out till satisfactory solution is reached. During the observation stage the component is investigated for various parameters like:

  - Fig. 3 Formability (FLD) i.e. cracks, wrinkles
  - Fig. 4 Thickness reduction-Final Thickness
  - Fig. 5 Effective Strain Distribution
  - Fig. 6 Effective Stress Distribution
  - Fig. 7 Displacement
  - Fig. 8 Deformation Mode

**B. Structural Analysis with Formability Result Mapping:**

HyperWorks Results Mapper is a HyperCrash based tool that provides a framework to initialize a structural model with results from a forming simulation.

Procedure involves loading the structural model and forming simulation results followed by mapping the results and finally exporting the mapped data in a structural solver format for Radioss/Abaqus. Mapping quantities include thickness, plastic strain, strain tensor and stress tensor [3].
After result mapping, structural analysis need to carried out for following loading condition, according to application engineering for component.

1) When pressing the Boot Can on the drive shaft a force of 392.4N (40 kg) is applied on it. It has to survive this load for 6 cycles of loading & unloading without deformation for correct fitment

2) When assembling the boot on the boot can, the crimping load applied is 196.2N (20 kg), it should avoid the deformation with this force for 100 cycles of loading & unloading

3) In operation condition pulling/sliding force of 147.15 N (15 kg) is acting on boot can by rubber boot. It has to survive for 150000 cycles of loading & unloading of this force without loosening of press fit between Boot Can & Shaft.

The basic steps for FE formability simulation of component and subsequent structural analysis with result mapping, are outlined schematically as in following figure.

Fig. 10The FE forming & structural simulation

### IV. RESULTS AND DISCUSSION

Several iterations of formability simulation for different coefficient of friction and blank holding force are carried out to analyse the results for maximum & minimum thickness of component after forming. Following table shows thickness results for different iterations. Optimized results are found at Coefficient of Friction= 0.2 & Blank Holder Force= 3 Tonnage.

<table>
<thead>
<tr>
<th>Run No.</th>
<th>Friction Coefficient</th>
<th>Blank Holding Force (Tonnage)</th>
<th>Blank Holding Pressure Level</th>
<th>Max Thickness (mm)</th>
<th>Min Thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.01</td>
<td>3</td>
<td>low</td>
<td>2.46</td>
<td>1.48</td>
</tr>
<tr>
<td>2</td>
<td>0.05</td>
<td>3</td>
<td>low</td>
<td>2.37</td>
<td>1.50</td>
</tr>
<tr>
<td>3</td>
<td>0.1</td>
<td>3</td>
<td>low</td>
<td>2.36</td>
<td>1.50</td>
</tr>
<tr>
<td>4</td>
<td>0.2</td>
<td>3</td>
<td>low</td>
<td>2.36</td>
<td>1.50</td>
</tr>
<tr>
<td>5</td>
<td>0.25</td>
<td>3</td>
<td>low</td>
<td>2.35</td>
<td>1.50</td>
</tr>
<tr>
<td>6</td>
<td>0.3</td>
<td>3</td>
<td>low</td>
<td>2.35</td>
<td>1.48</td>
</tr>
<tr>
<td>7</td>
<td>0.4</td>
<td>3</td>
<td>low</td>
<td>2.35</td>
<td>1.48</td>
</tr>
<tr>
<td>8</td>
<td>0.5</td>
<td>3</td>
<td>low</td>
<td>2.35</td>
<td>1.43</td>
</tr>
<tr>
<td>9</td>
<td>0.2</td>
<td>2</td>
<td>low</td>
<td>2.37</td>
<td>1.49</td>
</tr>
<tr>
<td>10</td>
<td>0.2</td>
<td>2.5</td>
<td>low</td>
<td>2.37</td>
<td>1.50</td>
</tr>
<tr>
<td>11</td>
<td>0.2</td>
<td>3.5</td>
<td>low</td>
<td>2.36</td>
<td>1.50</td>
</tr>
<tr>
<td>12</td>
<td>0.2</td>
<td>4</td>
<td>low</td>
<td>2.36</td>
<td>1.48</td>
</tr>
</tbody>
</table>

C. Effect Of The Blank Holding Force On The Wrinkling Of Cup Flange

To find the effect of the blank holder force on wrinkling of component Boot Can, study was carried out with some blank holder force values as 2, 2.5, 3, 3.5 & 4 tonnages, while coefficient of friction kept 0.2. The results indicated that when blank holder force is very low, the wrinkling is possible to occur more on component. As blank holding force increases, effect of wrinkles is reduced. The result shows that the blank holder force plays as important role in deep drawing for affecting the wrinkling, but it do not influent much on the thickness distribution of component.

D. Effect Of The Friction Coefficient On Wall Thickness Distribution

Result table shows the wall thickness distribution along with component centre line by using friction coefficient 0.1 to 0.25 is good than using friction coefficient < 0.05 and friction coefficient > 0.25, hence the friction coefficient 0.2 is considered as more suitable. The results found that the friction coefficient is one of the important parameters effects on the thickness of the component, but it more effective with suitable combination with blank holding force and other parameters.

V. EXPERIMENTAL VALIDATION

Experimentation is planned for the similar loading conditions of component in application; same as intended in structural analysis, on randomly selected components (up to 6) form the forming sampling lot of 200 parts.
Absence of strain/deformation hampering the fitment & serve purposes of component in drive shaft assembly is strong indication towards validation of the thesis of this project work. Component forming feasibility with lesser thinning than minimum required thickness is also important outcome of the work.

VI. CONCLUSION

The forming analysis was performed for different parameters and the best tryout was explained in this study. Since the forming process is carried out virtually using the CAE technology, we can predict the formability of the component without spending money on manual tryouts and corrections. Metal forming, product design & Die design industry can be largely benefited to carry the virtual forming simulation and thus reduce the manual tryouts which involves time and money. Simulation technique can be used effectively to optimize the product design and process parameters. Using HyperForm and available CAE technology, any modification required for the die or the component can be find out. Multiple iterations can be performed and accordingly the design can be finalized. To increase the accuracy and reliability of structural finite element simulations of sheet metal parts, it is essential to map forming analysis results to evaluate realistic structural integrity.

REFERENCES